

563.

**Experimental and Computational Study of
Trapped Vortex Combustor Sector Rig
With Tri-pass Diffuser**

**International Journal of Rotating Machinery
Special Issue Dedicated to the 70th Birthday of
Professor Wen-Jei Yang
The University of Michigan**

**R.C. Hendricks¹, D.T. Shouse², W. M. Roquemore²
D.L. Burrus³, B.S. Duncan³, R.C. Ryder⁴, A. Brankovic⁴, N. -S. Liu¹
J. R. Gallagher¹, J.A. Hendricks⁵**

- (1) NASA Glenn Research Center, Cleveland OH
- (2) Wright-Patterson Air Force Base, Dayton OH
- (3) General Electric Aircraft Engines, Cincinnati OH
- (4) Flow Parametrics, LLC, Bear DE
- (5) Diligent Design, Toledo OH

Correspondence: Robert.C.Hendricks@grc.nasa.gov

Robert C Hendricks
MS 5-9
NASA Glenn Research Center
21000 Brookpark Road
Cleveland, OH 44135
216 977 7507
216 433 3000 FAX

This report is a preprint of an article submitted to a journal for publication. Because of changes that may be made before formal publication, this preprint is made available with the understanding that it will not be cited or reproduced without the permission of the author.

Experimental and Computational Study of Trapped Vortex Combustor Sector Rig With Tri-pass Diffuser

**R.C. Hendricks¹, D.T. Shouse², W. M. Roquemore²
D.L. Burrus³, B.S. Duncan³, R.C. Ryder⁴, A. Brankovic⁴, N. -S. Liu¹
J. R. Gallagher¹, J.A. Hendricks⁵**

(4) NASA Glenn Research Center, Cleveland OH

(5) Wright-Patterson Air Force Base, Dayton OH

(6) General Electric Aircraft Engines, Cincinnati OH

(4) Flow Parametrics, LLC, Bear DE

(5) Diligent Design, Toledo OH

ABSTRACT

The Trapped Vortex Combustor (TVC) potentially offers numerous operational advantages over current production gas turbine engine combustors. These include lower weight, lower pollutant emissions, effective flame stabilization, high combustion efficiency, excellent high altitude relight capability, and operation in the lean burn or RQL modes of combustion. The present work describes the operational principles of the TVC, and provides detailed performance data on a configuration featuring a tri-pass diffusion system. Performance data include EINO_x results for various fuel-air ratios and combustor residence times, combustion efficiency as a function of combustor residence time, and combustor lean blow-out (LBO) performance. Computational fluid dynamics (CFD) simulations using liquid spray droplet evaporation and combustion modeling are performed and related to flow structures observed in photographs of the combustor. The CFD results are used to understand the aerodynamics and combustion features under different fueling conditions. Performance data acquired to date are favorable in comparison to conventional gas turbine combustors. Further testing over a wider range of fuel-air ratios, fuel flow splits, and pressure ratios is in progress to explore the TVC performance. In addition, alternate configurations for the upstream pressure feed, including bi-pass diffusion schemes, as well as variations on the fuel injection patterns, are currently in test and evaluation phases.

KEY WORDS : Combustion, CFD, Trapped Vortex Combustor, Emissions, Combustor Performance.

INTRODUCTION

The aircraft gas turbine combustor has been under continuous development and refinement for over fifty years, with dramatic improvements in specific fuel consumption, durability, reliability, size, weight and stability, and more recently, significant reduction of pollutant emissions such as CO and NO_x. The current generations of operational commercial and military engines are true marvels of high technology, equally as

impressive as the most advanced microchips or telecommunications devices available. Nevertheless, in the light of increasing international regulatory restrictions on emissions and noise, uncertainties in fuel prices, and demand for increased safety and reliability, research continues for improved combustion components and concepts, which will meet these conflicting demands.

The Trapped Vortex Combustor (TVC), pioneered by the Air Force Research Laboratory and General Electric Aircraft Engines (Ref. 1) and discussed herein represents a relatively simple and highly advantageous alternative to conventional gas turbine combustors. Combustion is stabilized within the trapped vortex cavity, allowing for simple liquid fuel injection techniques, and accurate control over fuel residence time, with resulting benefits to combustor stability, efficiency, and reduction of pollutant emissions. This is in contrast to swirl stabilized combustors, which require sophisticated fuel injector and swirler combinations to stabilize the flow aerodynamically in the primary combustion zone together with cooling and dilution air introduced through complex diffuser jet flows. These designs are susceptible to blow out, are difficult to relight, and perhaps most importantly, are difficult to design due to their fluid dynamic and combustion complexity. Designers engaged in this field are being challenged to develop schemes that meet current and near-term regulatory requirements for NO_x and greenhouse gas emissions using these designs.

The objective of this work is to develop the database of performance characteristics and operational behavior of the Trapped Vortex Combustor (TVC hereafter), with regard to application for advanced gas turbine combustor designs. This is done experimentally and through numerical simulations. Experiments are conducted on a full-size, planar sector rig with upper and lower cavities, fueled through a periodic, linear array of 10 pairs of cavity injectors. Combustor air is delivered through a triple-branch (or tri-pass) diffusion system, which is also fueled directly from simplex type pressure atomizing fuel injectors. The combustor can operate by fueling the cavity alone, or as a staged combustor system using a tri-pass main fueling and cavity fueling. Combinations over a wide range of fuel-air ratios and fueling splits between cavity and the main passages have been tested, with some results presented herein. Additional operational variables include combustor inlet temperature, pressure and pressure drop. The hardware was tested over a variety of fuel-air ratios, combustor inlet temperatures and pressures, with measurements taken of the exit temperatures, NO_x emissions, and lean blow-out (LBO) performance.

The experiments were simulated using a computational fluid dynamics (CFD) solver developed to compute multiphase, chemically reacting flows in complex geometries, such as those in aircraft gas turbine combustors. The FPVortexTM flow solver (a commercially available version of the National Combustion Code, or NCC) is applied to compute three cases of interest for the TVC test rig. Distinct fueling schemes were selected for the CFD runs, to illustrate differences in aerodynamic and combustion flow structures. Details of the CFD simulations are provided, and are discussed with regard to the photographic results obtained on the TVC rig at the same fueling conditions.

BACKGROUND

Trapped Vortex Combustor

The experimental apparatus used was an available 12" wide rectangular (planar) sector rig that was originally designed, fabricated and tested by a team consisting of representatives from Air Force Research Laboratory (AFRL) and GE Aircraft Engines (GEAE). This same group of AFRL and GEAE representatives participated with NASA GRC representatives to conduct this current experimental testing. The TVC combustor rig is a planar sector rig consisting of 10 modules arranged linearly. Opposed upper and lower cavities are present, all with liquid spray pressure atomizing fuel injectors, and numerous air jets, and film and slot cooling injections. The main combustor air is fed by a diffusion system consisting of three passages (tri-pass configuration), which split the incoming flow into approximately three equal flows. This flow enters the combustor at high, but subsonic speed. For the cavity only fueled operating mode, only upstream plenum air flow passes through the diffuser passages. For the cavity plus main operating mode, main stream fuel is supplied to each passage via a pair of opposed liquid spray pressure atomizing fuel injectors. In the process of fueling the main stream, the fuel is injected into the high-speed cross-streams passing through the diffuser passages to provide some degree of premixing of the main fuel and air before the mixture enters the combustion chamber for burning (see Refs. 1 and 2).

The 12" rectangular TVC sector rig hardware is shown in Figure 1. In this photograph one of the air cooled sidewalls has been removed for viewing. The arrow shown on the figure indicates flow from left to right. Station 3 (plenum) air enters into the diffuser, and is split into three streams by the diffuser struts at the upstream end of the tri-pass branches. The flow is diffused, however, it must wrap around a fuel strut perpendicular to the flow. The fuel strut is airfoil shaped, splitting the flow within each branch in half. Liquid spray simplex type pressure atomizing-fuel injectors are situated approximately midway down the fuel strut. When turned on, they inject directly into these tri-pass diffuser air streams. The upper and lower cavity fuel injection points (large openings on the cavity forward walls) can also be seen in the photo.

In the cavity only fueled operating mode high intensity combustion takes place and is largely confined to the cavity and the recirculation regions formed between the flows issuing from the tri-pass passages. For the cavity plus main operating mode, additional combustion takes place further downstream in the burner duct, as the liquid fuel from the main stream injection evaporates and is ignited by the hot products transported out of the cavity.

Photographic results for the three cases under study here are shown in Figures 2, 3 and 4. Case 1 results, taken at an inlet pressure of 345 kPa, inlet temperature of 533 K, overall equivalence ratio of $\phi = 0.53$, and fueling split between the cavity ($\phi = 0.65$) and main feed system ($\phi = 0.73$), are shown in Figure 2. Case 2 results, taken at inlet pressure of 345 kPa, temperature of 533 K, overall $\phi = 0.52$, and fueling split between the cavity ($\Phi=1.8$) and main ($\Phi=0.14$), are shown in Figure 3. Finally, cavity only fueling was used for Case 3, with cavity $\phi = 2.2$, and overall $\phi = 0.53$. These results are shown in Figure 4.

The rich fueling condition for the cavity results in highly intense combustion, with high fuel residence times, and overall high combustion efficiencies.

Computational Fluid Dynamics Methodology

The CFD modeling approach described in Ryder and McDivitt (Ref. 3) is followed in this work. An important aspect of this approach is to model the combustion system geometry precisely as it is installed on the test stand. The starting point in the process is development of a fully detailed computer aided design (CAD) solid model representation of the hardware. The CAD solid model is preferably based on the same CAD model as that used for structural and thermal finite element analysis, and also for manufacturing. This approach has the benefit of including not only all the major flow hardware features, but also additional and miscellaneous hardware such as fasteners, bolts, support brackets, and other assembly components which can often adversely affect the flow distributions and quality. A further benefit of this approach is that all flow paths which are aerodynamically coupled will be identified and be included in the model. Once a satisfactory model has been created, an "air-solid" representation of the hardware is created, using a Boolean operation. This is a solid model representation of the gas flow path. This model can then be automatically meshed using any of several commercial mesh generators, generally using tetrahedral elements. The resulting grid is tagged with boundary conditions, and output into an unstructured database read in by the flow solver. Intermediate results databases are created for assessing convergence, which can be viewed automatically in graphics post-processing packages. Further examples of this CFD methodology applied to gas turbine component flow fields may be found in the works by Ryder (Ref. 4) and Brankovic et. al. (Ref.5).

Physics-based modeling approaches are used to resolve the aerodynamics and combustion processes taking place within the combustion system. Turbulence is modeled using the two-equation or $k-\epsilon$ model of turbulence, together with wall functions. For combustion processes in which fuel-air mixing and flame stabilization are key phenomena to be resolved, turbulence wall functions provide adequate accuracy. If, however, pressure drop through the pre-diffuser, or heat transfer along liner walls are of concern, near-wall grid clustering together with wall integration models of turbulence are used instead of wall functions.

The combustion process is modeled through detailed tracking of the liquid fuel spray droplets, evaporation, and chemical reaction. Individual spray droplets are tracked using a Lagrangian trajectory algorithm, developed and described by Raju (Ref. 6). The droplet model includes physics models for droplet drag, heat and mass transfer to the liquid through established correlations. Droplet heating is modeled in detail, including the effect of internal circulation within the droplet, and temperature variations within the droplet. Droplet initial conditions are obtained from fuel injector experiments, and droplet initial velocity, radius and temperature fields are specified for each injector. In the present case, the orifice injectors have been measured to produce 90-degree conical spray droplet flow, and these results were used as the cavity and the main injector fueling initial conditions. Discrete liquid droplet streams are clocked at 10-degree intervals, originating at the external lip of the fuel injector orifice. The droplet size distribution is modeled using a

correlation taken from El Banhawy and Whitelaw (Ref. 7), applied to each droplet stream. The liquid spray droplet model is fully coupled to the aerodynamics and combustion models in the flow solver through continuity, momentum and energy source terms. Typically, the smaller diameter droplets evaporate and react first, followed by the larger droplets, which penetrate further, into the flow field.

A CFD simulation of the speed contour provides a convenient overview of the main aerodynamic features of the TVC with tri-pass diffuser system, and is shown in Figure 5. The plenum conditions for the CFD simulations to be discussed differ from those in the photos in terms of the air mass flow rates through the diffuser and the secondary flow (or cooling air) paths. In the experimental runs shown in the photos, the tested mass flow rate resulted in peak flow velocities corresponding to $Mach = 0.7$ within the diffuser branches. In the CFD simulations, plenum conditions were run at a mass flow rate that produced a peak $Mach = 0.3$, also in the diffuser branches. This difference is expected to impact the computed temperatures within the combustor. For example, at the higher Mach number flow rate run in the experiment, shear velocity between the cross flow and the liquid fuel droplet would be greater, resulting in a higher fuel evaporation rate, and consequently, more rapid combustion and heat release. In the CFD simulations at the lower diffuser Mach number, lower shear and lower droplet evaporation rates would be anticipated. As such, some fuel would be still bound in the liquid droplets, and left the domain unreacted. At these lower combustion efficiencies, bulk temperatures within the combustor were anticipated to be lower than in the experiment.

In the aerodynamics definition sketch, the box drawn around the combustor indicates the region shown in the photographs in Figures 2, 3 and 4. The upper figure is cross A-A offset from the combustor center plane, and clearly shows the interactions of the tri-pass diffuser airflows, with the cavity flow, with subsequent mixing in the downstream region. Air jets issuing into the upper and lower cavities are clearly seen, with rapid bending of the jets due to strong interactions with the combustion flow fields. Large-scale recirculation regions can be seen between the upper, middle and lower branches of the tri-pass diffusion system. These recirculation regions hold hot products of combustion, which enter along the cavity back wall. These recirculation zones form exceptionally stable combustion regions, and are aerodynamically stable over a wide range of fuel-air ratios and inlet temperatures and pressures.

The lower figures show cross sections of the combustor taken at three axial positions within the cavity: near wall (1-1), mid-plane (2-2), and downstream wall (3-3), as indicated by the cross section cuts above. The high-speed jets are clearly observed, as are the liquid fuel injection and the air jets issuing from the plenums. Although not easily observable in these views, significant amounts of slot and film cooling airflows are also entering into the cavity.

RESULTS

Comparison of Combustor Simulations with Photographs

CASE 1 (Run 2311) : Main and cavity fueling. In Case 1, fuel was injected into both the main tri-pass branches and into the TVC cavities. The TVC cavity equivalence ratio was $\phi = 0.65$, the main stream equivalence ratio was $\phi = 0.73$, and the overall system equivalence ratio was $\phi = 0.53$, based on a summation of all airflow in the cavity, the main stream, and downstream sections. The experimental photograph for the case is shown in Figure 2, representing a time- and path-averaged spanwise view through all 10 sectors of the combustor. The thermal-flow structure is represented in the photo through the luminosity of the flame. While not directly proportional to temperature, the observed flame luminosity shows high intensity burning within the cavity, particularly along the cavity back wall, as well as the central portion of the combustor. The flow simulation results for Case 1 are shown in Figure 6, in terms of cross sections through the combustor center plane (Figures 6 a through d), and axial cuts through the cavity (Figures 6 e through g).

The model calculation confirms that combustion is intense within the cavity, and particularly along the center plane, as seen in Figures 6 (a) and (b). Hot products of combustion are entrained from the cavity region into recirculation regions between the tri-pass diffuser branches, and also along the downstream wall of the cavity, which wrap around a 90-degree turn, mixing with the film cooling flow. In addition, with significant fueling into the main tri-pass diffuser passages, some combustion takes place in the near field of those flows. Most of the fuel entering through the main stream reacts further downstream in the straight duct, as seen by the hot streaks in Figure 6. Strong mixing in the lateral or span wise direction takes place away from the center plane, as seen in Figures 6 (c) and (d), with well-mixed high temperature flow throughout the straight duct. Interaction of the hot combustion gases with the liner film cooling and slot flow is clearly observed along the upper and lower portions of the combustor duct, flushed by a cool layer of fresh air.

The cross sections through the TVC cavity in Figures 6 (e) through (g) give further detail of the combustion flow structure. Near the cavity back wall, seen in Figure 6 (e), there is strong fluid coupling of the fuel jet with the upper, central air jet issuing from the plenum. This acts to enlarge the effective combustion area of the liquid jet stream. Figure 6 (e) also shows the hot flow evolving from the outer tri-pass legs, indicated by the two hot spots. These are due to rapid combustion of the fuel as it enters the combustion chamber. Interestingly, the center passage of the tri-pass diffuser flow features no combustion at this plane, but instead, that flow experiences a significant combustion delay until that fuel-air mixture reaches the downstream duct. These overall features can also be observed in the luminosity photo of this case in Figure 2. There is clearly a cooler zone in the central region, corresponding to the issuing of the center passage air-fuel stream, followed by a delayed combustion zone further downstream in the burner duct. Distinct combustion flow structures are seen in Figure 2, including a hot streak due to flow from the upper tri-pass branch together with entrained flow from the cavity, and also the hot central jet which eventually reacts and combusts quickly once ignited. A distinct cool region also exists in the upper and lower right corners of the cavity, both in the experimental results, and in the computational simulation. Also, intense burning along the back cavity wall is seen in the experiment and computational simulation.

CASE 2 (Run 2318): Reduced main fueling, increased cavity fueling. In Case 2, fuel was again injected into both the trapped vortex cavity and through the tri-pass diffuser passages, but in different ratios. Inlet pressure and temperature were the same as in Case 1. The cavity was richly fueled, with local equivalence ratio of $\phi = 1.8$, the main stream local equivalence ratio of $\phi = 0.14$, and the overall system equivalence ratio of $\phi = 0.52$. The experimental photograph for the case is shown in Figure 3, while the CFD simulation results for this case are shown in Figure 7, in the same format as the cross sections in Figure 6.

The combustion processes in both the cavity and in the central combustion zone are more varied than in Case 1. In the cavity, the rich fueling results in a strong interaction of the liquid fuel jet with the opposite cavity wall, with that fuel reacting quickly and intensely with incoming air flow from the cavity aft wall driver holes. The hot products of combustion associated with this interaction are purged rapidly into the downstream combustor section, and interact with the slot and film cooling flow along the combustor duct. There is an indication that hot cavity flow comes in contact with the upper and lower cavity walls, despite significant film cooling and air jet flow. There is also a suggestion in the photo in Figure 3 that this occurs in the rig as well.

The combustion process in the central combustor zone is also more varied than in Case 1. Larger combustion recirculation regions are present between the tri-pass diffuser outer and center passage flows, with hot flow also engendered from the tri-pass passages due to direct fueling. The shear layer formed between the cavity and tri-pass diffuser outer branch supports a high temperature gradient, separating the driven cavity flow and the high speed, slightly heated tri-pass jet flow. Rapid mixing of the combustion gases from the various sources, and with the film and slot cooling flows, occurs in the combustor duct further downstream.

CASE 3 (Run 2142): Cavity fueling only. In Case 3, fuel was injected into the trapped vortex cavity only, with no fuel injected into the diffuser passages. For this case, the cavity was richly fueled at a local equivalence ratio of $\phi = 2.2$, while overall system equivalence ratio was maintained at $\phi = 0.53$. Inlet pressure and temperature were again the same as in Case 1. The experimental photograph for Case 3 is shown in Figure 4, while the CFD simulation results for the case are shown in Figure 8, in the same format as for the previous cases.

The combustion process for this case forms a highly intense combustion zone that originates entirely within the cavity. As in the other cavity flow fields, the flame is attached to the edges formed by the spray cone, and anchored along the periphery of the hole near the injector orifice. The flame is turned by the various jet and film cooling flows, and wraps around the tri-pass airflows, filling the recirculation regions between the outer, center and inner passages of the tri-pass diffuser. These regions are also very hot, with products of combustion entrained from the cavities and the incoming tri-pass diffuser air flows. As in the previous cases, hot products mix out well in the downstream burner duct.

Combustor Experimental Performance Data

To establish the engineering performance parameters required for evaluation of the combustor, data were obtained for NO_x emissions, combustion efficiency, and lean blow-out, at different inlet temperatures, different fuel-air ratios, and different cavity to main stream fuel splits. The majority of the measured data were obtained in the cavity only fueled operating mode. The measured NO_x emissions have been corrected to standard day humidity levels.

Results for humidity-corrected EINO_x versus burner bulk cold flow residence time are shown in Figure 9. These results indicate that a factor of 3 reduction in the cold flow bulk residence time can reduce NO_x emissions by 33% to 50%. The observed 50% reduction in NO_x emissions is in line with anticipated benefits of using TVC technology in advanced military gas turbine engine applications such as the F414-400 used on the Navy F18 E/F aircraft.

Combustion efficiency versus bulk cold flow residence time is shown in Figure 10. High values of combustion efficiencies (>99%) are achieved over almost the entire range of residence times, over different fuel-air ratios, and cavity to main fuel splits. The effect on measured combustion efficiency is significantly amplified for overall lean operation ($f/a = 0.012$) at low operating inlet temperature (500 F).

Overall burner exit equivalence ratio at lean blow-out (LBO) performance versus burner cold flow reference velocity is shown in Figure 11. Values of 0.1 are generally considered good performance for conventional swirl-stabilized combustion devices which typically operate at reference velocities of 50 ft/sec or less. The TVC generally performs at or below this value at reference velocities up to a factor of 3 higher, with most data points between the 0.06 and 0.08 levels. Based on prior test experience with this prototypical TVC test rig it was anticipated that LBO performance would be exceptionally good for this device.

SUMMARY AND CONCLUSIONS

This paper described experimental testing and numerical simulation of a Trapped Vortex Combustor rig, with a view towards understanding the fundamental aerodynamic and combustion processes under different fueling conditions, and evaluation of engineering performance factors such as EINO_x combustion efficiency and lean blow-out. Photographs of the combustor hardware and photographs of the observed flame luminosity under three differing cavity to main stream fueling splits led to insights into flame structure and potential hot spots within the combustion zone. CFD simulation results for the liquid fuel spray droplet evaporation and combustion process showed in detail the complex aerodynamic flow interactions, and captured the very different combustion processes at work in the combustor main and cavity zones. It was noted that the CFD runs were performed at a lower diffuser Mach number, with reduced droplet evaporation rates and, consequently, reduced overall combustion efficiency than in the experiments. Additional experimental testing and data reduction showed the baseline

TVC configuration with tri-pass diffuser to be very promising with respect to key performance parameters, and in comparison to values for conventional, swirl-stabilized combustors. The TVC combustor is continuing to undergo testing at AFRL, and is being systematically and vigorously tested over a wider range of fueling and inlet conditions. Additional design work is also underway in tailoring the cavity and main stream fuel injection features, in ways that will enhance fuel-air mixing and further improve performance characteristics of the TVC class of combustors.

ACKNOWLEDGEMENTS

The authors would like to thank Dr. Marvin Goldstein NASA Glenn Research Center Strategic Research Fund, the Air Force Research Laboratory (AFRL) and GE Aircraft Engines (GEAE) for their assistance in making this research possible.

REFERENCES

1. Roquemore, W.M., Shouse, D., Burrus, D., Johnson, A., Cooper, C., Duncan, B., Hsu, K-Y., Katta, V.R., Sturgess, G.J. and Vihinen, I., 2001. Trapped Vortex Combustor Concepts for Gas Turbine Engines. AIAA Paper 2001-0483. 39th AIAA Aerospace Sciences Meeting and Exhibit, 8-11 January 2001, Reno, Nevada.
2. Burrus, D.L., Johnson, A.W., Roquemore, W. M., and Shouse, D.T., 2001. Performance Assessment of a Prototype Trapped Vortex Combustor Concept for Gas Turbine Application, ASME IGTI Paper 2001-GT-0087, ASME IGTI Turbo Expo 2001, 4-7 June 2001, New Orleans, Louisiana.
3. Ryder, Jr., R.C., and McDivitt, T., 2000. Application of the National Combustion Code Towards Industrial Gas Fired Heaters, AIAA Paper 2000-0456.
4. Ryder, Jr. R.C., 2000. Application of the National Combustion Code Towards Unsteady Mixing and Combustion Modeling, AIAA Paper 2000-0335.
5. Brankovic, A., Ryder, Jr., R.C., Sturgess, G.J., Lee, J., Kushari, A., Lubarsky, E., Zinn, B.T., 2001. Computational and Experimental Study of Aerodynamics and Heat Release In a Liquid Fueled Combustor, AIAA Paper 2001-0976.
6. Raju, M.S., 2000. An Overview of the NCC Spray / Monte Carlo PDF Computations, AIAA Paper 2000-0337.
7. El Banhawy, Y. and Whitelaw, J.H., 1980. Calculation of the Flow Properties of a Confined Kerosene-Spray Flame", AIAA J., Vol. 18, No. 12, pp. 1503-1510.

Flow
direction

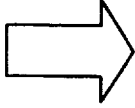


Figure 1. Test apparatus photo for the Trapped Vortex Combustor with Tri-Pass diffuser system, featuring a rectangular array of 10 fuel injector modules in a planar combustor sector rig. The near-field side wall has been removed for visualization. The fuel injectors can be seen on the rear plate of the cavity. Carbon deposits are visible on the ceramic combustor coating, reflecting extensive hot-fired testing. Film cooling holes can be seen along the cavity walls, combustor core, and combustor liner walls.

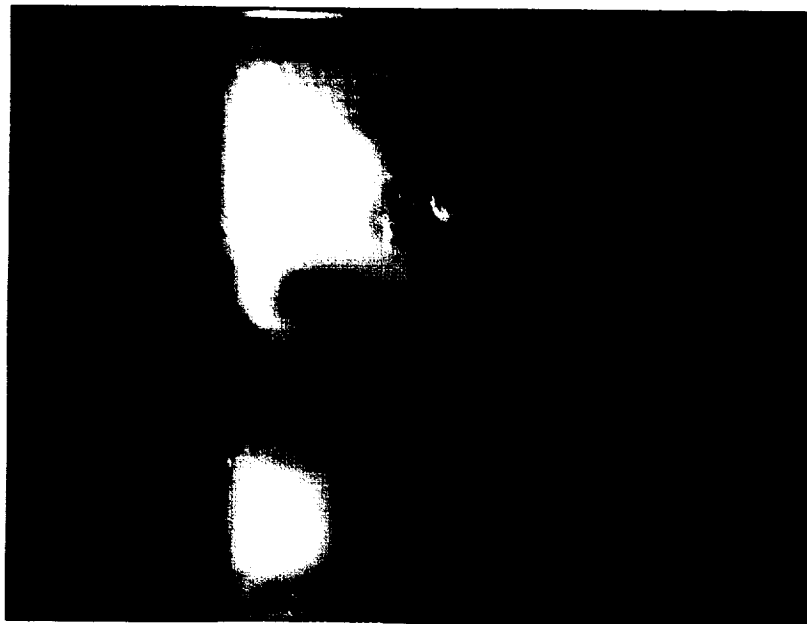


Figure 2. Photo of the Trapped Vortex Combustor with Tri-pass diffuser, Run 2311.

Inlet pressure 345 kPa (50 psia); fueling into cavity and main feed arms. Cavity $\phi = 0.65$, main feed arms $\phi = 0.73$, overall $\phi = 0.53$. $T_3 = 533$ K (500 F).

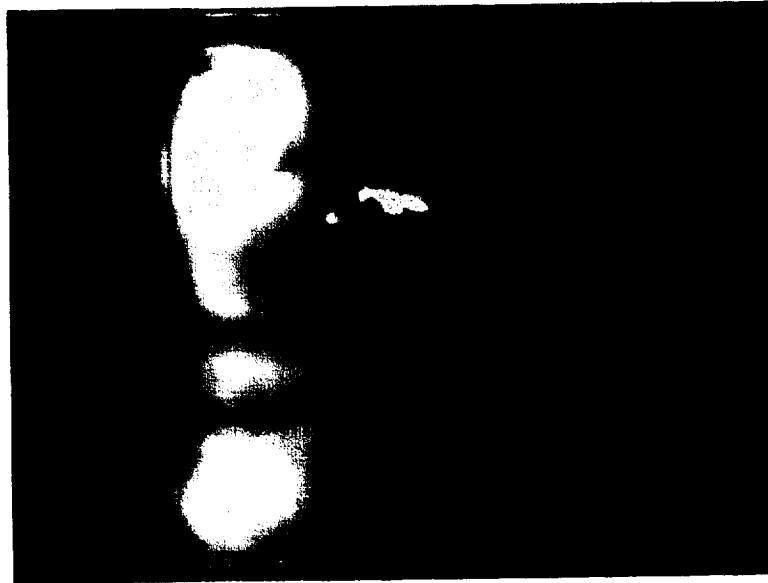


Figure 3. Photo of the Trapped Vortex Combustor with Tri-pass Diffuser, Run 2318.
Inlet pressure 345 kPa (50 psia); fueling into cavity and main feed arms. Cavity $\phi = 1.8$, main feed arms $\phi = 0.14$, overall $\phi = 0.52$. $T_3 = 533$ K (500 F).

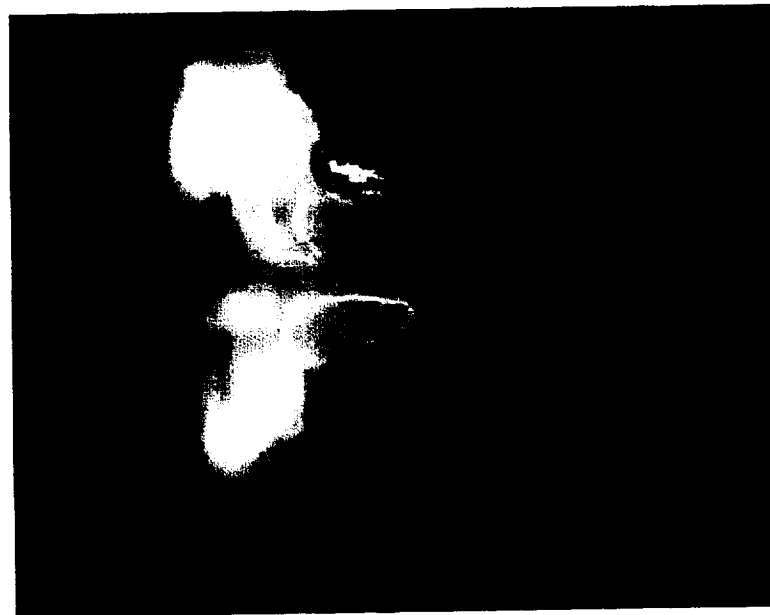


Figure 4. Photo of the Trapped Vortex Combustor with Tri-pass Diffuser, Run 2142.
Inlet pressure 345 kPa (50 psia); cavity fueling only. Cavity $\phi = 2.2$, overall $\phi = 0.53$. $T_3 = 533$ K (500 F).

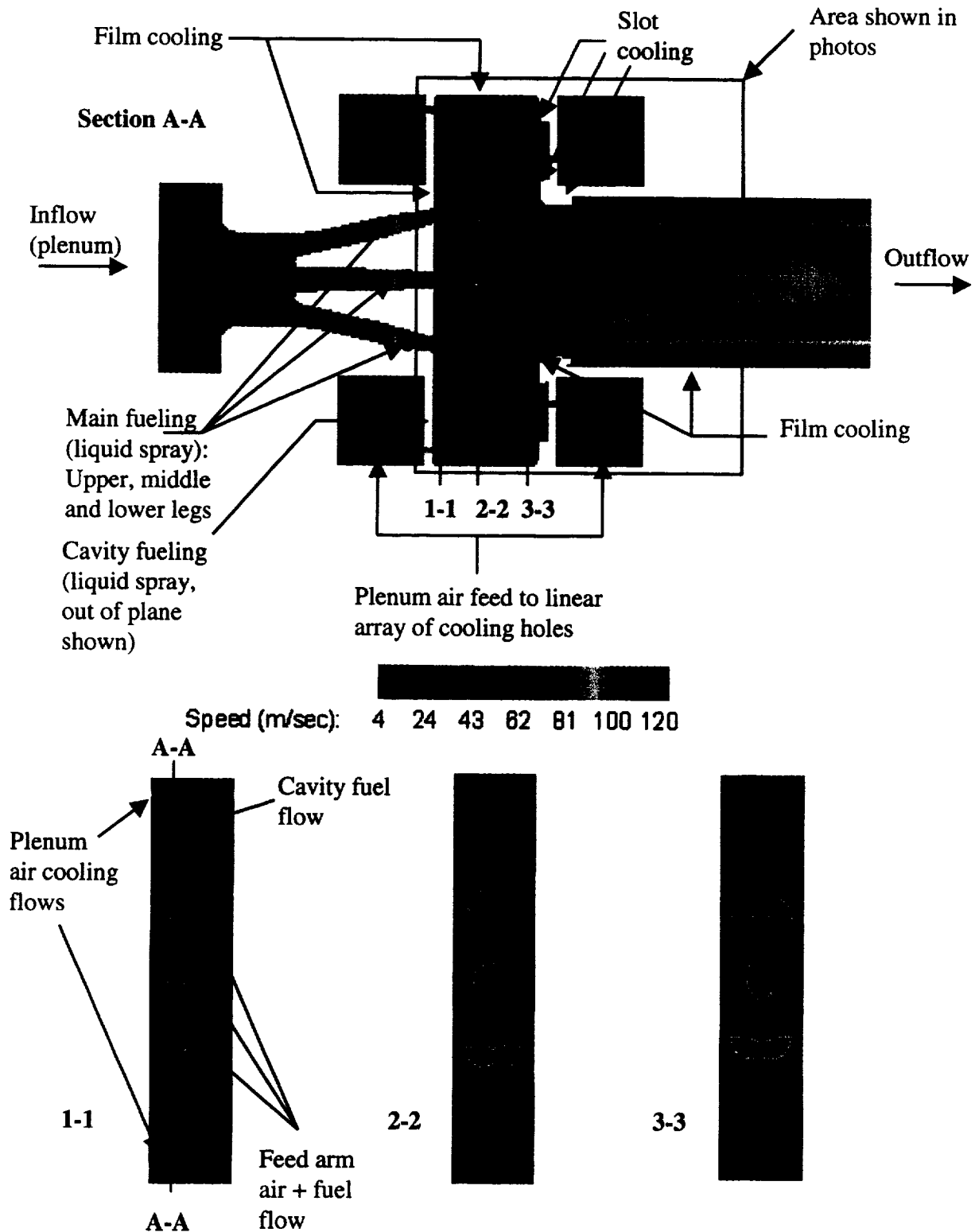


Figure 5. Speed contours and definition sketch for CFD simulation of the Trapped Vortex Combustor with Tri-pass diffuser, showing air and fuel flows. Upper figure is a cross-section taken midway between combustor center-plane and end-wall. Lower figures represent cuts at three planes within the cavity.

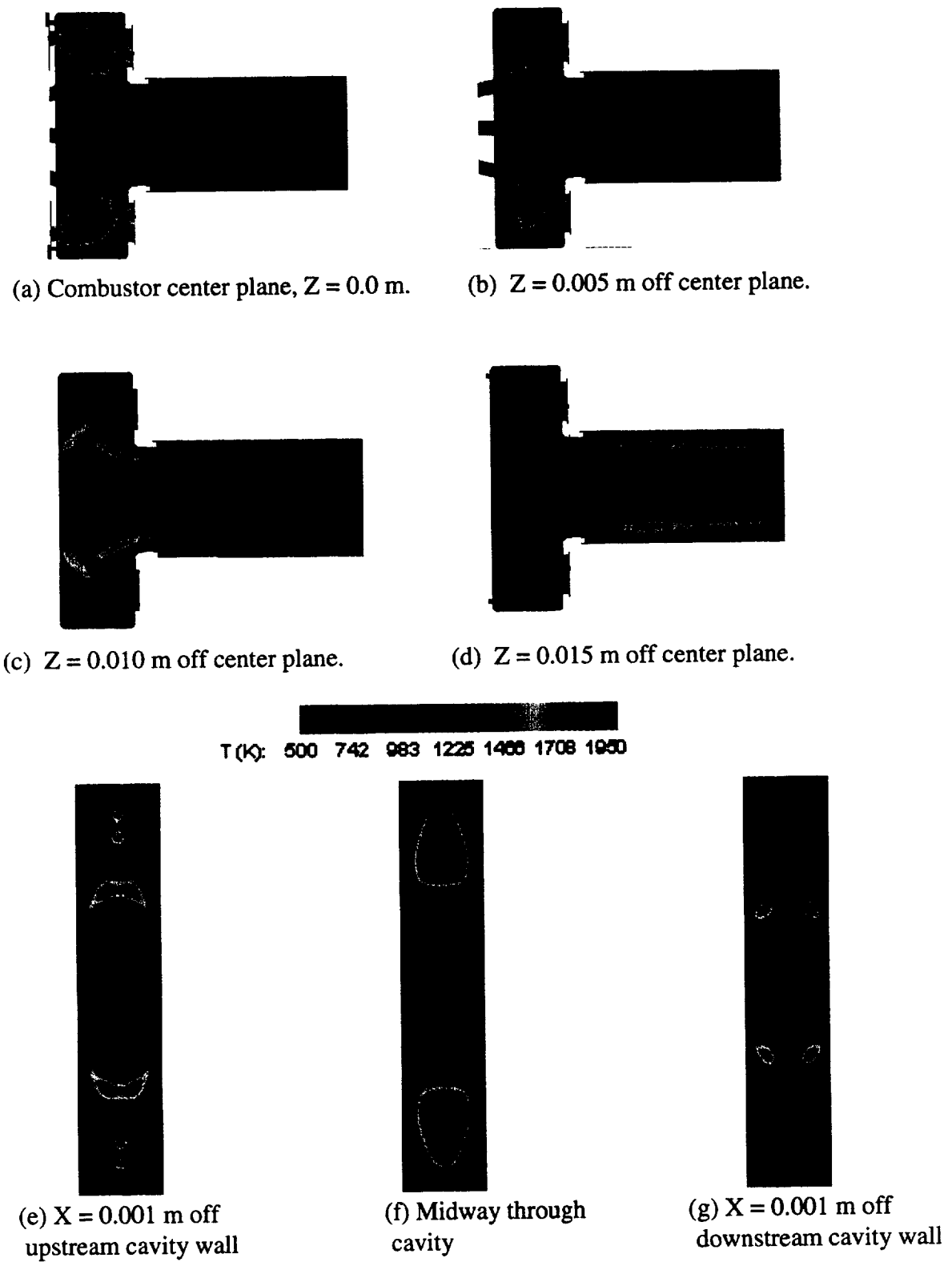


Figure 6. Computed temperature contours for Case 1. (a) - (d) Lengthwise cuts through the combustor. (e) - (g) Axial cuts through cavity.



(a) Combustor center plane, $Z = 0.0$ m.



(b) $Z = 0.005$ m off center plane.



(c) $Z = 0.010$ m off center plane.



(d) $Z = 0.015$ m off center plane.

T (K): 500 742 983 1225 1466 1708 1950



(e) $X = 0.001$ m off upstream cavity wall



(f) Midway through cavity



(g) $X = 0.001$ m off downstream cavity wall

Figure 7. Computed temperature contours for Case 2. (a) - (d) Lengthwise cuts through the combustor. (e) - (g) Axial cuts through cavity.

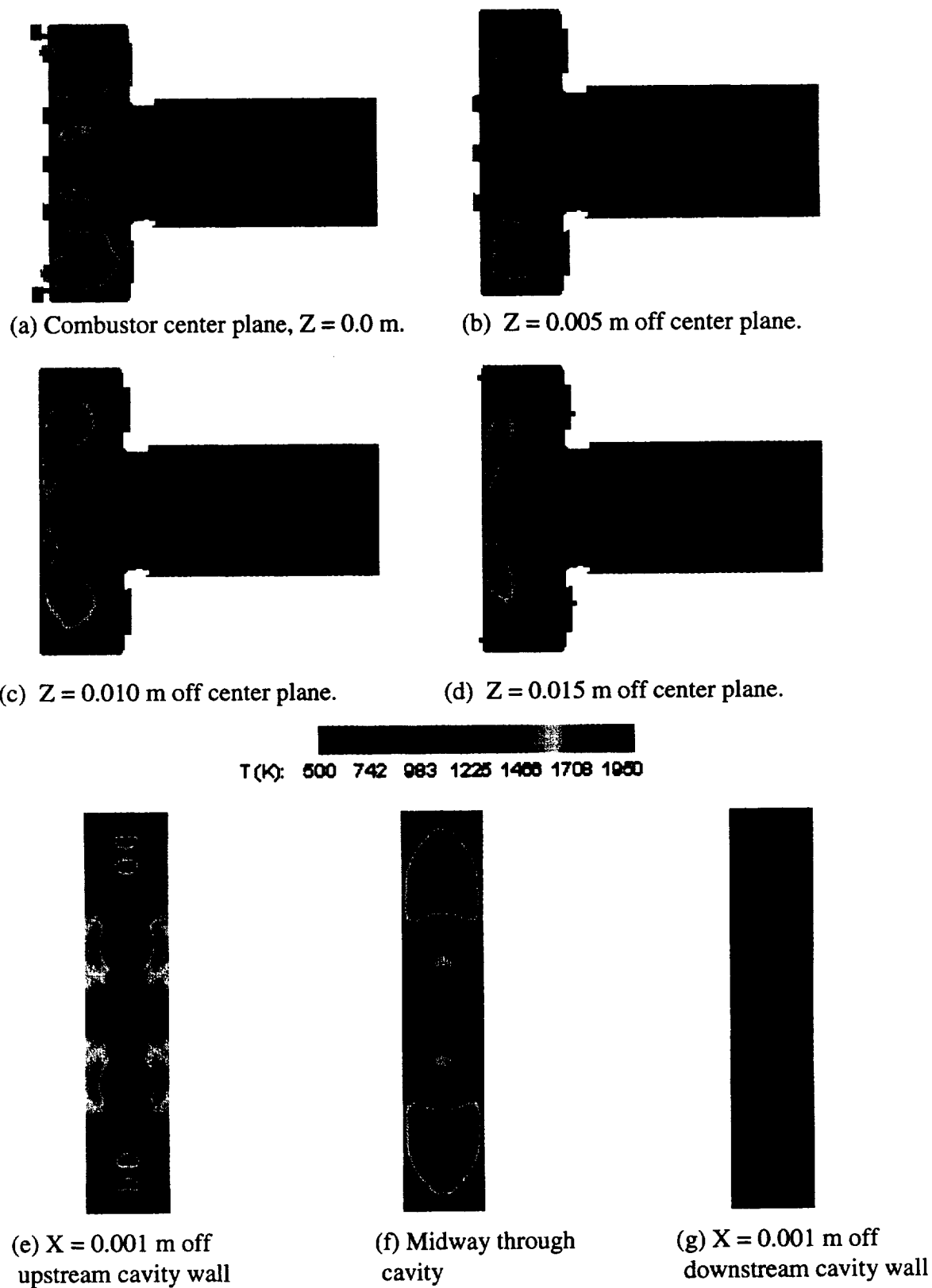


Figure 8. Computed temperature contours for Case 3. (a) - (d) Lengthwise cuts through the combustor. (e) - (g) Axial cuts through cavity.

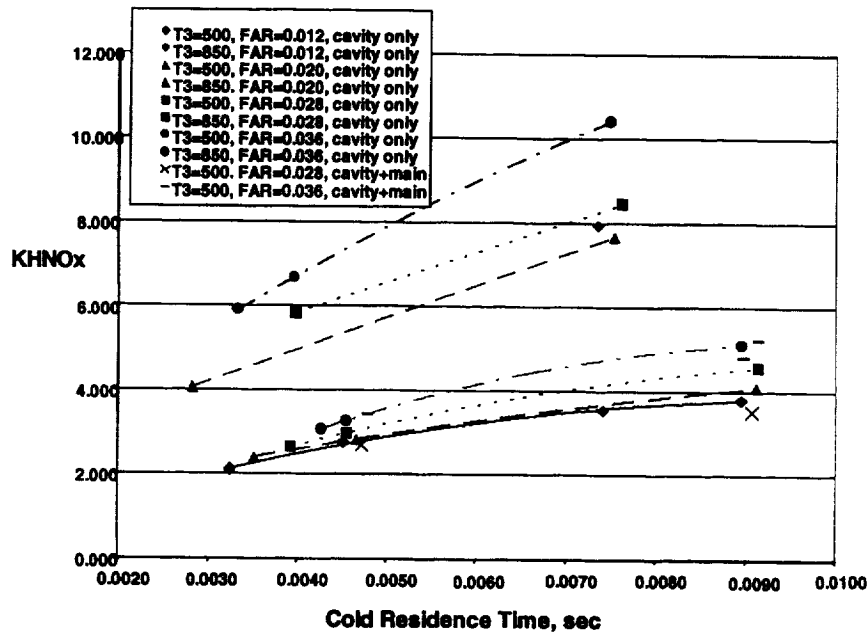


Figure 9. Experimental TVC-Tri-pass data of humidity-corrected EINOx vs. burner bulk cold residence time, at various test conditions. Inspection of the plot reveals that NOx can be reduced from 33% to 50% of baseline values, and that residence times can be controlled over a range of 2 to 3 times. NOx is seen to be relatively insensitive to T_4 . Pressure drop performance at P_4 is about 20% to 25% of P_3 . Stable combustor operation observed over all parameter settings.

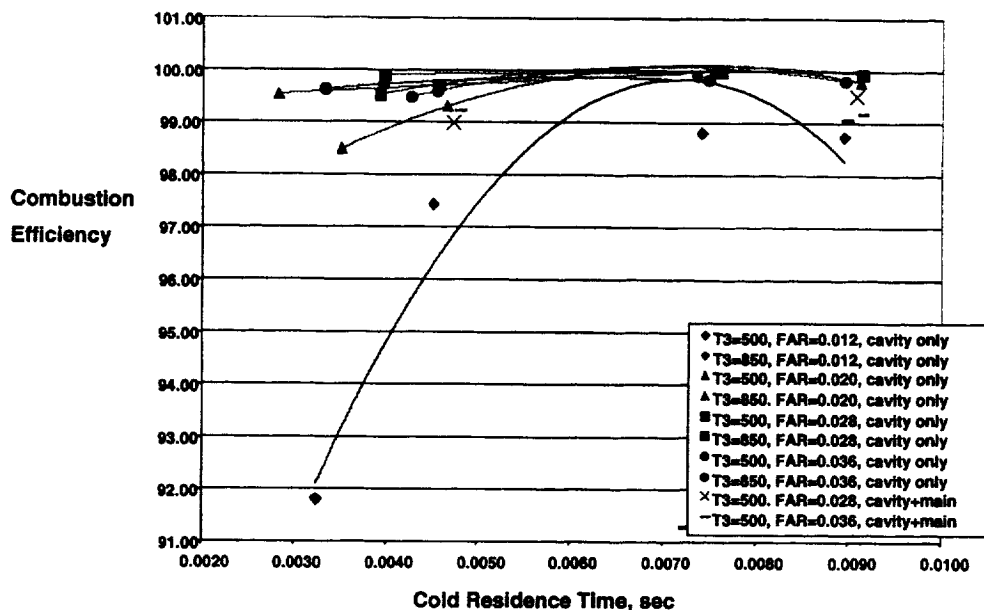


Figure 10. Experimental TVC-Tri-pass data of combustion efficiency vs. bulk cold residence time. Combustion efficiency decreases with residence time, significantly at low fuel-air ratios. for example, less than 98.5% for $f/a < 0.02$. Further measurements planned for the performance region of $0.012 < f/a < 0.02$.

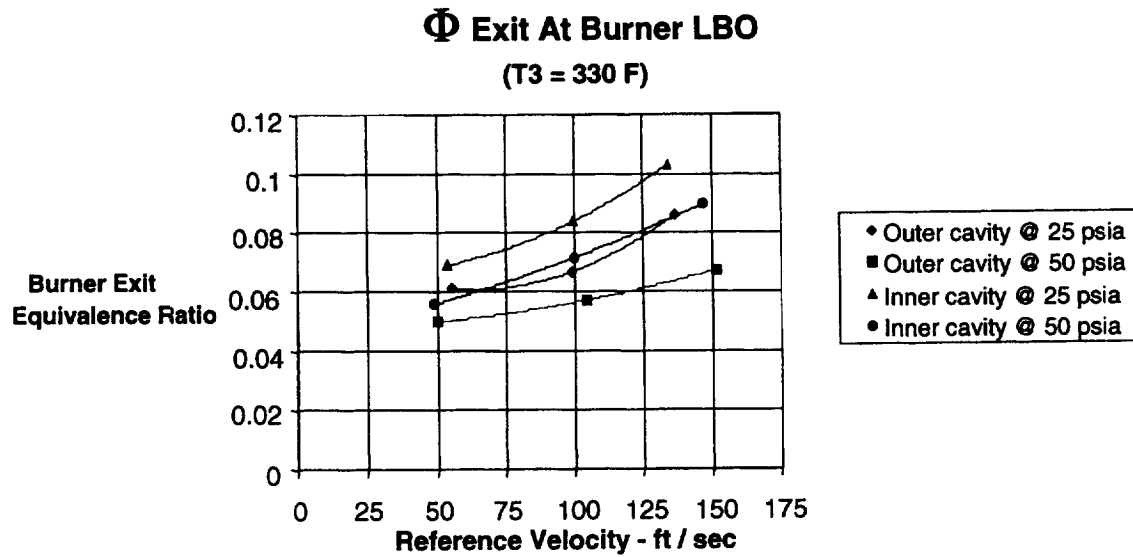


Figure 11. Experimental TVC-Tri-pass data of combustor lean blow-out (LBO) vs. reference velocity, at 172.5 kPA (25 psia) and 345 kPA (50 psia). For this case, cavity $\phi = 0.28$, overall $\phi = 0.359$. A value of 0.1 is considered good performance for conventional gas turbine combustor LBO performance. Trend observed is that LBO increases as reference velocity increases.

